Steady State Lid Driven Cavity Using Simple Algorithm - Finite Volume Method

Ganesh Borde

December 2024

Abstract

This project focuses on the numerical simulation of flow inside a two-dimensional lid-driven cavity. The study investigates the flow properties at low Reynolds numbers to achieve a steady-state solution. A pressure correction approach, specifically the Semi-Implicit Method for Pressure Linked Equations (SIMPLE), is employed to solve the incompressible Navier-Stokes equations. The finite volume discretization (FVM) method is used. The numerical results are validated against benchmark solutions of cavity flow reported in the literature. Various flow characteristics and results are analyzed in detail.

1 Introduction

The lid-driven cavity is a benchmark problem in the study of Computational Fluid Dynamics (CFD). Its simplicity allows researchers to evaluate the accuracy and stability of numerical methods. This problem has been extensively studied for over 50 years. Let us discuss about the problem below.

1.1 Problem description

The lid-driven cavity is a two-dimensional domain bounded by three stationary walls and one moving boundary, i.e., the top wall moving with a velocity U, as shown in Fig. **??**. The cavity is considered a square domain, where the length of all sides is L. As the top wall moves with a velocity U, the flow inside the cavity experiences motion due to the viscous nature of the fluid.

The velocity values at the three fixed boundaries are $u = v = 0$, while the moving boundary has $u = U$ and $v = 0$. In this project, the cavity is studied by setting the velocity $U = 1\,\mathrm{m/s}$ and the length of the boundary $L = 1\,\mathrm{m}$. Only the viscosity of the flow is varied, while the density of the flow is kept constant. The flow properties are analyzed for different viscosities. The pressure fields are not included in the results section, as their sole purpose in this study is to project the velocity field onto the solenoidal space.

1.2 Governing equations

As mentioned, the flow is steady, incompressible, two-dimensional and isothermal. Therefore, the 2-D incompressible Navier-Stokes equations are considered, which are divided into the continuity equation, x-momentum equation, and y-momentum equation.

After non-dimensioning the variables as:

- Length scale $x = \overline{x}L$ and $y = \overline{y}L$
- Velocity scale $u = \overline{u}U$ and $v = \overline{v}U$
- Pressure $p = \overline{p}\rho U^2$

The resulting governing equations (omitting the bar notations for the sake of simplicity) are:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \tag{1.1}$$

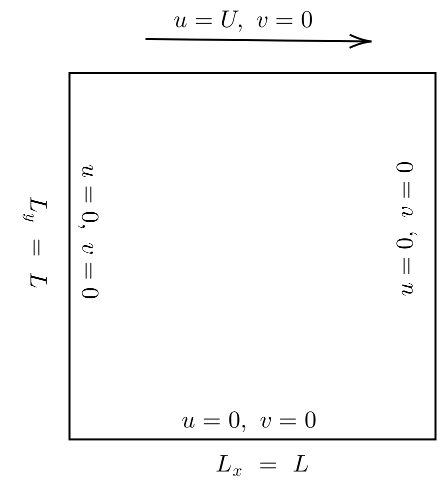

Figure 1.1: Problem description of lid-driven cavity flow.

$$\frac{\partial uu}{\partial x} + \frac{\partial uv}{\partial y} = -\frac{dp}{dx} + \frac{1}{\text{Re}}\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right], \tag{1.2}$$

$$\frac{\partial uv}{\partial x} + \frac{\partial vv}{\partial y} = -\frac{dp}{dy} + \frac{1}{\text{Re}}\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right]. \tag{1.3}$$

In Eq. **??** and **??**, Re is reynolds number of the flow, defined as:

$$\text{Re} = \frac{\rho U L}{\mu}, \tag{1.4}$$

where μ is the viscosity of the fluid.

1.3 Literature review

Lid-driven cavity flow problem is a test case type problem for incompressible solvers. The past work done on lid-driven cavity flow gives a range for steady solution. For a wide range of Reynolds numbers, Shen [**?**] examined the flow structures in a unit cavity and discovered that the flow establishes a steady character for Re 10000.

The difference between the projection method and the SIMPLE method is the projection method is generally second-order accurate, which is developed for the simulation of incompressible unsteady flows by employing a non-linear update of pressure term, which may depend on the grid size, time step, and even velocity. It has three and four-step projection method. One of the famous methods by Chorin .[**?**].The standard SIMPLE method is written in a concise formula for steady and unsteady flow. It is proven that SIMPLE type methods have second-order temporal accuracy for unsteady flows. The classical second-order projection method and SIMPLE type methods are united within the framework of the general second-order projection formula .[**?**].

2 Numerical method

This section deals with the numerical method, discretization, and grid approach in solving the lid-driven cavity. The SIMPLE algorithm on the staggered grid configuration is used. First, let's talk about the staggered grid.

_grid.png

Figure 2.1: Staggered grid configuration

Table 1: The number of variables in the domain

Field quantity	Interior resolution	Resolution with boundary points
p	$n_x \times n_y$	$(n_x + 2) \times (n_y + 2)$
u	$(n_x - 1) \times n_y$	$(n_x + 1) \times (n_y + 2)$
v	$n_x \times (n_y - 1)$	$(n_x + 2) \times (n_y + 1)$

2.1 Staggered grid

A staggered grid is a configuration used for spatial discretization. Scalar variables (such as pressure p, density ρ, total enthalpy h, etc.) are stored at the cell centers of control volumes, whereas velocity components (such as u and v) or momentum variables are stored at the cell faces. This configuration differs from a collocated grid, where all variables are stored at the same locations. For compressible or incompressible flow simulations using structured grids, staggered storage is primarily utilized.

Collocated grids are prone to a discretization error known as odd-even decoupling, which leads to checkerboard oscillation patterns in the solutions. By using a staggered grid, odd-even decoupling between pressure and velocity can be effectively avoided. In a staggered grid, pressure values (p) are located at the center of the cells, while the velocity components u and v are stored at the cell faces, as shown in Fig. **??**.

Boundary conditions can be defined in two different forms: Dirichlet boundary condition, where primary variables are specified, and Neumann boundary condition, where derivatives, such as pressure gradients, are specified.

To apply boundary conditions, ghost cells (cells beyond the actual computational domain) are introduced. Ghost cells in the x-direction are represented by dashed lines, which indicate the v-velocity, while those in the y-direction represent the u-velocity. Pressure nodes are considered in both the x- and y- directions. The updated number of variables in the system is presented in Table **??**.

2.2 Discretization

The governing equations are discretized using finite volume method. The velocity at cell center p, is approximated by cell centers of east e, west w, north n, and south s. The u is indexing at (i, j) and v v at (I, J). So, the u momentum equation (Eq. **??**) can be approximated as

$$\frac{((uu)_e-(uu)_w)\Delta x}{+}\frac{((uv)_n-(uv)_s)\Delta y}{=}-\frac{p_{I+1,j}-p_{I-1,j}\Delta x}{+}\frac{1}{Re}\left[\frac{u_{i-1,j}-2u_{i,j}+u_{i+1,j}}{(\Delta x)^2}+\frac{v_{I-1,J}-2u_{I,J}+u_{I+1,J}}{(\Delta y)^2}\right], \tag{2.1}$$

where u_e,u_w,